Image Cover Sheet

CLASSIFICATION	SYSTEM NUMBER 508296
UNCLASSIFIED	

TITLE

IMPROVED SHIP DETAIL FINITE ELEMENT STRESS ANALYSIS

System Number:

Patron Number:

Requester:

Notes:

DSIS Use only:

Deliver to:

Improved Ship Detail Finite Element Stress Analysis

Neil G. Pegg*, **David Heath***, **Mervyn E. Norwood**♦

*Defence Research Establishment Atlantic, P.O. Box 1012, Dartmouth, Nova Scotia, Canada, B2Y 3Z7

♦Martec Ltd., Suite 400, 1888 Brunswick St., Halifax, Nova Scotia, Canada, B3J 3J8

Abstract

This paper describes an improved approach to finite element modelling of ship structural details for design and in-service assessment. Specialized meshing algorithms have been developed which create finite element meshes of three-dimensional detail structure from predefined templates of the detail boundaries and associated meshing parameters. The ship structural details are divided into classes such as stiffener intersections, brackets, bulkhead-stiffener intersections or cutouts, which would have a common set of meshing parameters. Classes can be identified from lists of standard ship structural details produced by classification societies or for a specific ship type. Parametric stress analyses can be undertaken for these classes in advance to determine suitable meshing parameters which can be stored in a database and used to quickly generate the required detail meshes when required. The overall time to produce a ship detail mesh is reduced from several days using general purpose finite element model generators to a couple of hours.

This improved detail meshing approach is being incorporated into an overall finite element based ship structural analysis program which uses global ship displacement results in a semi-automated top-down analysis of structural details. Specialized spectral sea load codes are used with the global ship model to produce stress spectra or extreme loads in the details for fatigue or ultimate strength analysis. Overall, this approach greatly reduces the time necessary to undertake complex analysis of ship structural details.

Introduction

In their continuous efforts to improve safety and cost efficiency, classification societies and Navies have been developing 'first-principles' rational assessment methods for ship structures [1,2,3,4]. These computer methods are based on numerical modelling of the sea loads (Figure 1), which the vessels are likely to see in different operations, and of the resulting structural behaviour with the finite element method (FEM). Finite element analysis (FEA) is required to obtain reasonable estimates of the hull girder response and of the complex stress patterns occurring in structural details such as cutouts and connections. The rapid advances in computing power have made it possible for an engineer to undertake large scale FEA (10^5 degrees of freedom) in a reasonable time period on inexpensive computer hardware. The top-down approach, which uses coarser mesh models of the complete ship hull structure to

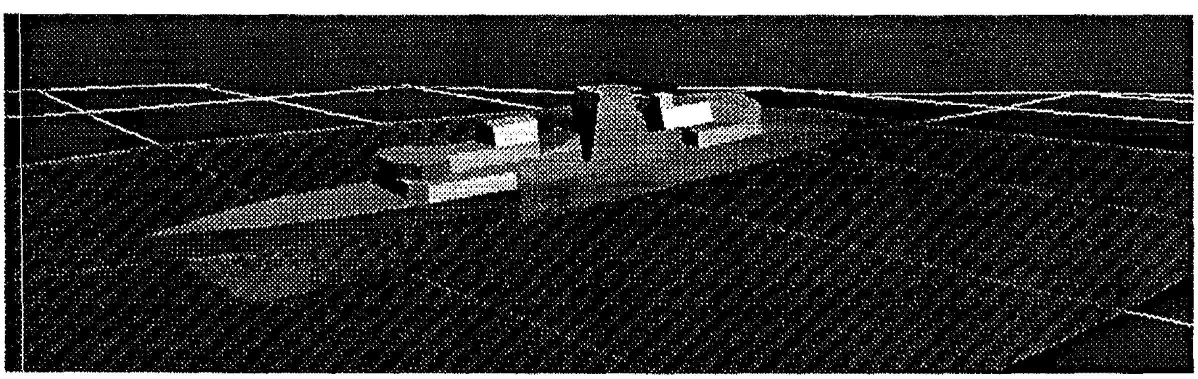

Figure 1: Computer modelling of sea loads and structural response for complete ship

provide boundary loads to fine mesh models of structural details, has become an accepted approach in many of the rational ship structural analysis tools.

While the computation time for the large finite element (FE) models has become acceptable, there remain some major drawbacks to applying the FEM to routine design and analysis of ship structure. The first is the amount of time required to create the FE models. Even the more advanced general purpose FE mesh generators require significant time and skill to produce models of complex three dimensional shell structure. The second main problem, which is not addressed to any significant extent in this paper, is how to use the results of FE analysis in assessment of the strength and endurance of the structure.

This paper discusses an approach to FE meshing of ship structural details which will significantly decrease the time required to undertake FEA to the point where it can be undertaken in a routine and timely manner. The topic of ship structural detail meshing classes will be discussed followed by a description of a specialized FE mesher that can be used to create FE meshes of the detail classes. Approaches to modelling of damage such as cracks and corrosion and methods of assessing the effects of this damage are then proposed. Several examples of this approach are then given.

Top-Down Ship Structural Detail Analysis

The analysis method used for this work is top-down FEA of the global ship model and the local detail area of interest. The global ship model is a coarse mesh Maestro [5] model of either the entire ship or a longitudinal section. Loads are applied as pressure and inertia forces from three-dimensional sea load codes, and/or as sectional bending moments and shear forces. The detail FE analysis is done with the general purpose code VAST [6]. The detail meshing and interface between the various programs, and of the top-down data, is done by the program MGDSA (Maestro Graphics Detail Stress Analysis). The top-down interface is undertaken by automatically identifying nodes that are common to the Maestro global model and the detail VAST model. These become master nodes on the boundaries between the two models. Other nodes, which may be created on the boundary of the detail model, are automatically slaved to the master nodes to give the correct translation of displacement boundary conditions between the two models. Of particular value is the facility to automatically create translational and rotational boundary conditions when going from a beam element representation of a stiffener in the global model to a shell element representation in the detail model. In addition to creating the detail meshes and the top-down analysis data, the MGDSA code also has facilities to verify the models and plot displacement and stress results. Current developments of this code include connections to other FE packages such as ANSYS and NASTRAN, an integration of an ultimate strength analysis code and a variety of sea load codes, and a new version of the code using an improved object-oriented data management system.

Ship Structural Detail Classes

The initial developments for detail meshing in MGDSA were based on approaches similar to those used in general purpose FE modellers. While the overall integration of the top-down analysis and development of detail models greatly enhanced the ability to undertake detail FEA of ship structure, the time to produce the detail FE meshes is still often prohibitive. In order to overcome this drawback, an approach to meshing ship structural details is being implemented which will reduce the time to hours instead of days and make routine FEA of details a possibility.

Ships are usually built to a defined set of details which are documented by classification societies, owners (such as Navies), or the builders. Details include stringer and frame intersections, bulkhead and stiffener connections (watertight and non-watertight), penetrations, cutouts, etc. The defined detail may vary in component dimensions (plate thickness, web height, etc.) and possibly have minor variations such as the inclusion of tripping brackets. Detail can thus be defined by a set of 2D patches which are defined by boundary lines (this approach uses interior as well as exterior boundary lines to include cutouts and crack lines) and connected together at common boundaries to form the 3D detail. The 2D patches are meshed based on a description of the node distribution on the boundary lines consistent with a

special purpose FE mesher (discussed below). Parametric stress analysis of the detail classes is undertaken to produce the best predefined set of nodal distributions for meshing the details. The nodal distributions may be defined by functions of the detail geometry as opposed to fixed values (eg. a function of plate thickness or cutout dimension). Development of the detail classes and the parametric

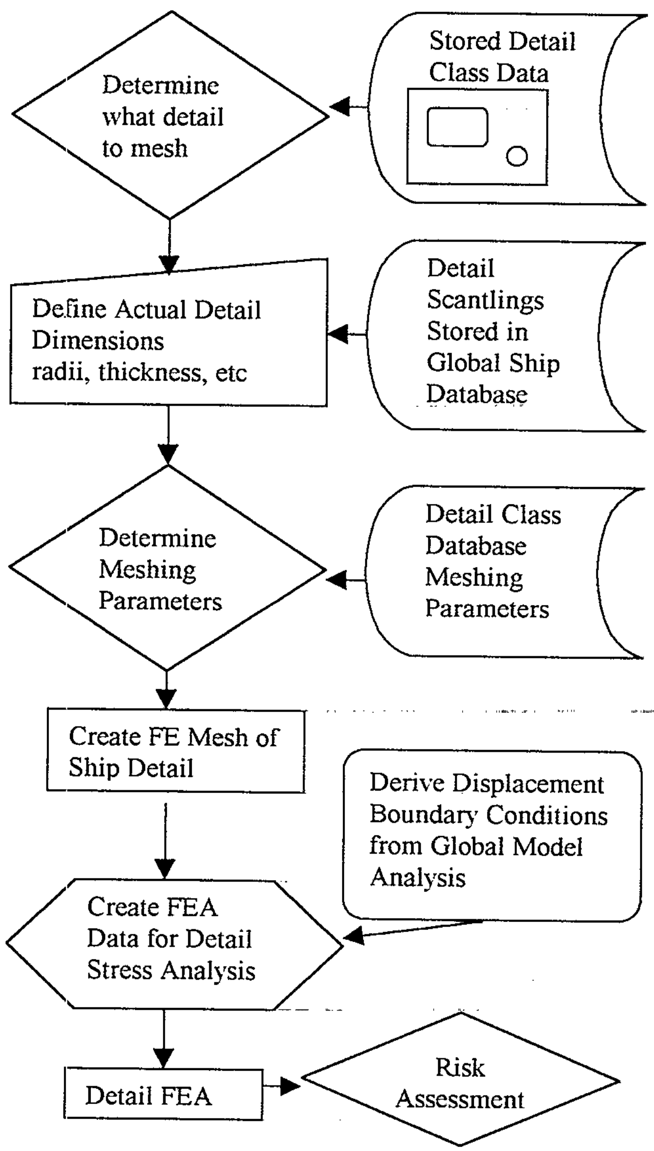

Figure 2: Schematic of top-down analysis with detail class FE mesh data

analysis will take considerable time, and is to be done before use in a specific ship analysis. The detail class descriptions and meshing parameters are stored in a database that can be called during a ship analysis to quickly generate the detail FE mesh of interest. As the object-oriented database management system is developed, the specific detail classes will be identified at their locations in the global ship model and include the actual scantling information for that detail. The overall approach to top-down detail meshing is illustrated in the flowchart of Figure 2.

Currently, 30 structural detail classes are being developed for a Canadian frigate. Figure 3 shows a standard longitudinal stringer and frame or beam intersection divided and meshed in modelling planes. In this case there are 8 modelling planes; 2 for each flange (4), 1 for each web (2) and 2 for the plating of the shell. This particular model stops at the frame but could extend through the other side creating an additional number of planes. Each different geometric plane in Figure 3 is bounded by a set of boundary lines. The overall dimensions of the planes and the boundary lines of this class can vary to allow a variety of stiffener dimensions to be accommodated.

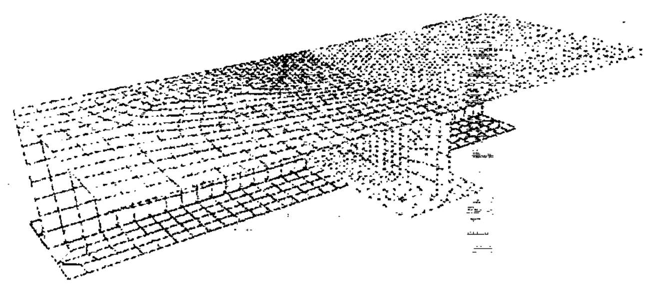

Figure 3: A detail 'class' of a longitudinal/frame intersection showing the modelling planes

Specialized Detail Finite Element Mesher

A meshing algorithm has been developed which uses a paving method to produce a mesh of quad elements on any two-dimensional surface made up of any number of boundary lines. This includes interior (multiple cutouts, crack lines) as well as exterior boundaries. The boundary lines are described by both geometric primitive shapes and node distribution algorithms. Each two-

dimensional surface describes one part of a detail class. Figure 4 shows some examples of meshes created by the detail finite element mesher. Work is still underway in testing and improving the algorithm. Special features to control symmetry and limit element size as a function of their location in the mesh domain are being developed and implemented. Assembling the two dimensional planes together produces the three-dimensional models. Compatability between the planes is ensured by using the same boundary line definition at the common boundaries in the meshing process. Curved surfaces are handled by first mapping the curved surface on to a flat surface for meshing and then mapping the element and node distribution back to the curved surface.

Modelling of Structural Damage

The Canadian Department of National Defence's (DND) main interest in developing a first principles computer based approach to analysis of ship structure is to assess the effects of in-service structural damage on ship operational capability. The tool will also be used to assess options for design changes during major refits. This is being done within an overall project entitled ISSMM (Improved Ship Structural Maintenance Management)[7] which is producing computer tools to aid in developing a more efficient repair and maintenance process. The primary questions to be answered are:

⇒ Given the detected damage, does it have to be repaired now or can it wait?

⇒ What operational limits should be placed on the damaged ship to ensure safety?

⇒ What is the most cost effective repair strategy?

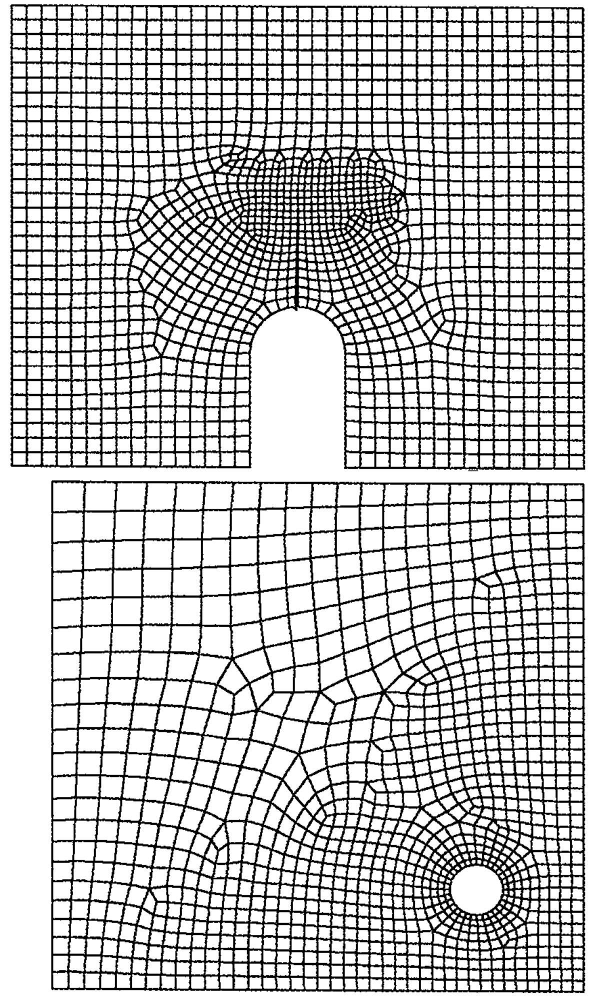

Figure 4: Examples of 2-D meshes created by the specialized detail mesher (a hatch cutout containing a crack propagating upwards and a plate with an arbitrarily placed and sized hole)

In-service damage can consist of corrosion, cracks or deformations from collisions. The tool will also be extended to consider weapons damage. In order to effectively use the ISSMM tool, the modelling and assessment of the detected damage has to be done in a very short timeframe. The database of the global ship and the structural detail classes will be in place to quickly create models of the area of interest and produce loads for a variety of operating conditions. Additions to the finite element mesher are being developed which will automatically create generic or user defined patterns of corrosion, fatigue cracks or deformation patterns.

The effects of corrosion and deformation will be assessed through changes in stress states over the undamaged structure for the same loads. Assessment of the local ultimate strength of the damaged component will also be undertaken through a semi-automated nonlinear FEA of stiffened panel components

(integrated structural unit method). The damaged stiffened panels will be used to produce new load-shortening curves for full global hull cross section ultimate strength analysis.

For crack growth, the complete detail mesher being developed in this program will have the ability to mesh a crack and crack tip located anywhere in the detail of interest (as illustrated in Figure 4). It will be possible for the crack to be progressed according to a particular crack growth law and automatically remeshed to provide new crack tip stress intensity results. Once the detail finite element mesh has been developed and a connection is made to the global model, a series of representative load cases would be run to develop a stress history for crack initiation or crack growth analysis. The Canadian DND is funding development of a series of codes entitled LIFE3D [8] which encompass various crack initiation and crack growth models to be used in conjunction with FEA stress results. The global FE model has also been used to undertake spectral analysis directly by coupling it to a frequency domain sea loads code [9,10]. Stress or stress intensity spectra at the detail of interest resulting from this process will be used in the crack growth or crack initiation models. A question which is being studied at this time, is where to take stress values for crack initiation analysis. The detail class mesh has to be designed to give suitable stress values for the initiation models and material parameters that are being used. A hot-spot stress approach is being investigated which uses models that produce stress in the region of a typical weld-toe location.

Evaluation of the damage results will be undertaken by both deterministic analysis through assessment of the change in safety factor, and probabilistic analysis using limit states for stress, fatigue and ultimate strength.

Examples of Improved Detail Meshing

Refit of Large Penetrations in Ship Bottom Structure

During a recent refit of DND's research vessel CFAV QUEST, several large penetrations (the largest being approximately one meter in diameter) were placed in the midship bottom structure to house sonar instrumentation. MGDSA was used for the analysis of this structural modification [11]. Figure 5 shows the detailed model overlayed on part of the global ship model and the stress results around one of the holes.

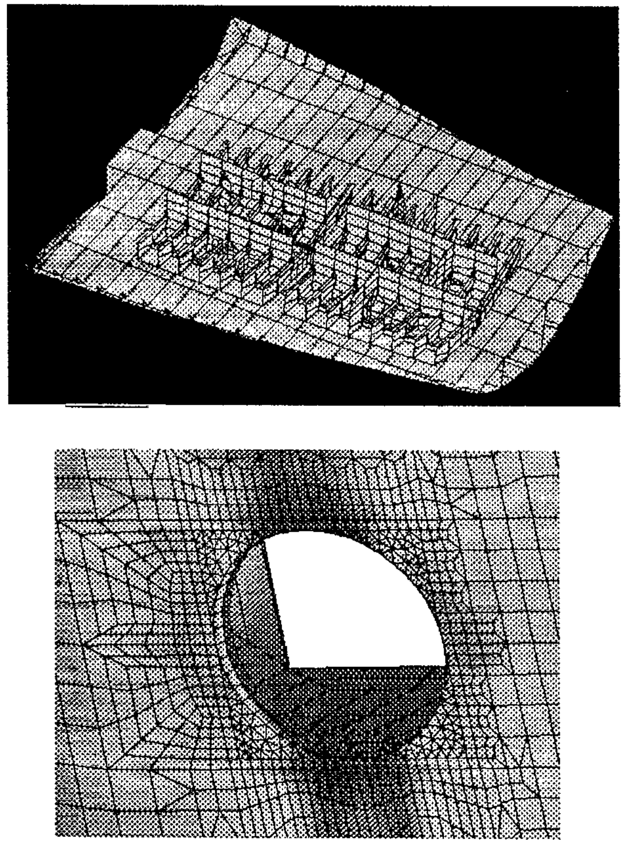

Figure 5: MGDSA detail modelling of large penetrations in ship bottom structure

CPF Main Deck Frame/Stringer Intersection

One of the main types of structure which requires detail meshing is the intersection of transverse frames and longitudinal stiffeners. Figure 6 shows a portion of a global Maestro model of midship bottom and sideshell structure including the frames and some longitudinals. MGDSA has a special feature to automatically convert the MAESTRO strake elements into refined meshes of stiffened panels. This is illustrated in the coarser mesh region of the second model in Figure 6. The specialized

detail mesher was then used to create the mesh of the intersection shown in the central part of the second model in Figure 6. Also included in Figure 6 are the stress results for the detail.

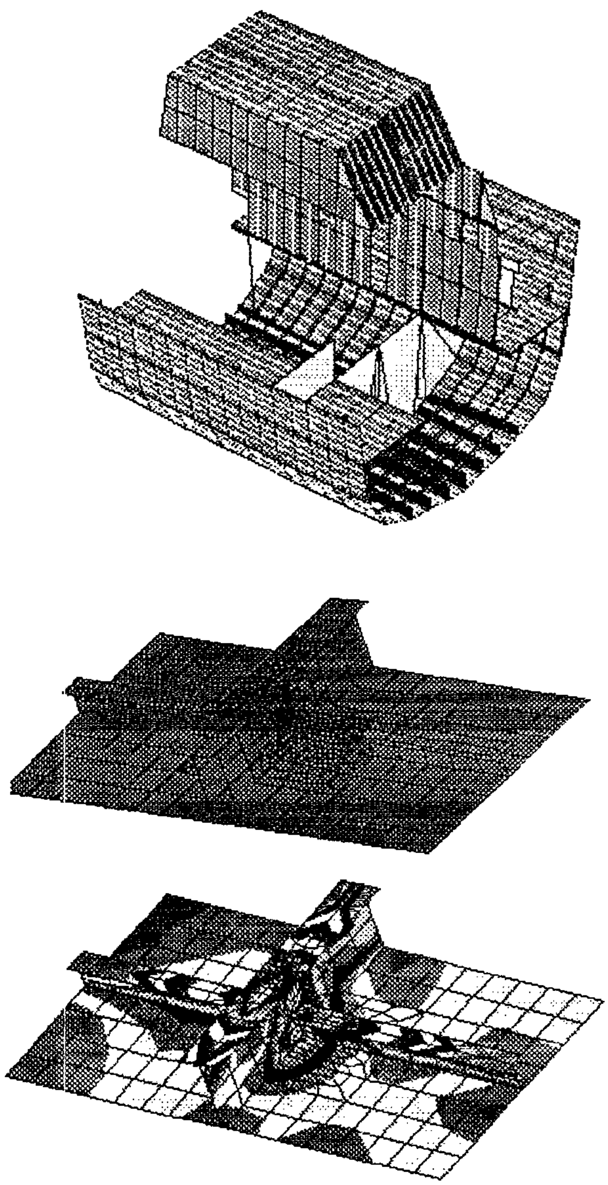

Figure 6: MAESTRO model and MGDSA detail model of a stiffener/frame intersection

Crack Propagation at Main Deck Penetration

A feature of the specialized mesher is its ability to implement a crack into detail structure. The crack line is treated as an internal boundary line in the meshing plane. VAST has special crack-tip elements which are used in the vicinity of the crack tip to give the stress intensity values which are used in fatigue crack growth laws. Figure 7 shows a crack detail created in a longitudinal bulkhead from the Maestro model (the top model of Figure 6), where the crack (outlined in red) is propagating from a deck cutout. The resulting mesh and stress contour plot is shown in Figure 7.

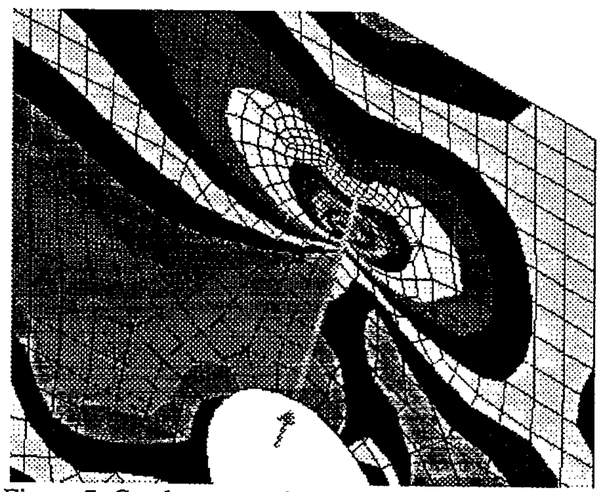

Figure 7: Crack propagating from a bulkhead penetration

Conclusions

At the time of writing, the overall method of detail analysis described in this paper is under development. Most of the components have been developed in an initial state and are being integrated and tested against realistic ship structural analysis scenarios. Many of the components of the proposed method are currently being applied by users of Maestro through the MGDSA program.

The rapid advances in computing power have made it possible to apply extensive numerical computations in routine engineering analysis. The challenge now is to make these methods easily accessible to the users. This requires the development of integrated analysis programs and data management tools and presentation

of results to the user in an easily understood and reliable manner. The subjects discussed in this paper are directed towards this end and have demonstrated what can be expected for routine analysis of ship structures in the future.

References

1. 'Veristar, Integrating Design And Analysis And Ship Management', Bureau Veritas program documentation, 1997.
2. 'Safehull, Technical Information', American Bureau of Shipping, 1997.
3. 'Nauticus, The Classification Information Highway', Det Norske Veritas Classification AS, 1995.
4. 'Shipright, Design, Construction And Lifetime Ship Care Procedures', Lloyd's Register Report 95/50/3.11, 1998.
5. MAESTRO - Method for Analysis Evaluation and Structural Optimization, User's Manual - Version 7.0', distributed by Proteus Engineering Ltd, Annapolis, MD, July 1992.
6. VAST - Vibration and Strength Finite Element Code, User's Manual - Version 7.0, Martec Ltd., 1996.
7. Pegg, N.G., Gibson S., "Application of Advanced Analysis Methods to the Life Cycle Management of Ship Structures", Advances in Marine Structures III, DERA, Scotland, 1997.
8. Wallace, J.C., and Chernuka, M.W., 'LIFE3D Three-dimensional Crack-growth Simulation Program Suite User's Manual', Martec Limited, July 1990.
9. Stredulinsky D.C., 'Proposed Method for Calculation of Spectral Response to Random Wave Loading Using Top-Down Finite Element Modelling', DREA Technical Memorandum 97/209, November, 1997.
10. 'Finite Element Based Spectral Analysis Methods for Fatigue Crack Growth and Initiation of a Ship Operating in a Seaway', Martec Limited, September, 1997.
11. Pegg N.G., 'Finite Element Stress Analysis of Midlife Refit Penetrations in QUEST Bottom Structure' DREA Technical Memorandum 97/249, September 1997.

#508296